SVFLUX™

2D / 3D Seepage Modeling Software

Tutorial Manual

Written by:
Robert Thode, B.Sc.G.E.

Edited by:
Murray Fredlund, Ph.D., P.Eng.

SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada

Software License

The software described in this manual is furnished under a license agreement. The software may be used or copied only in accordance with the terms of the agreement.

Software Support

Support for the software is furnished under the terms of a support agreement.

Copyright

Information contained within this Tutorial Manual is copyrighted and all rights are reserved by SoilVision Systems Ltd. The SVFLUX software is a proprietary product and trade secret of SoilVision Systems. The Tutorial Manual may be reproduced or copied in whole or in part by the software licensee for use with running the software. The Tutorial Manual may not be reproduced or copied in any form or by any means for the purpose of selling the copies.

Disclaimer of Warranty

SoilVision Systems Ltd. reserves the right to make periodic modifications of this product without obligation to notify any person of such revision. SoilVision does not guarantee, warrant, or make any representation regarding the use of, or the results of, the programs in terms of correctness, accuracy, reliability, currentness, or otherwise; the user is expected to make the final evaluation in the context of his (her) own problems.

Trademarks

Windows™ is a registered trademark of Microsoft Corporation.
SoilVision® is a registered trademark of SoilVision Systems Ltd.
SVOFFICE™ is a trademark of SoilVision Systems Ltd.
SVFLUX ™ is a trademark of SoilVision Systems Ltd.
CHEMFLUX ™ is a trademark of SoilVision Systems Ltd.
SVSOLID ™ is a trademark of SoilVision Systems Ltd.
SVHEAT ™ is a trademark of SoilVision Systems Ltd.
SVAIRFLOW ™ is a trademark of SoilVision Systems Ltd.
SVSLOPE ™ is a trademark of SoilVision Systems Ltd.
ACUMESH ™ is a trademark of SoilVision Systems Ltd.
FlexPDE® is a registered trademark of PDE Solutions Inc.

Copyright © 2008
by
SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada
ALL RIGHTS RESERVED
Printed in Canada

1 Introduction

The Tutorial Manual serves a special role in guiding the first time users of the SVFLUX software through a typical example problem. The example is "typical" in the sense that it is not too rigorous on one hand and not too simple on the other hand.

The Tutorial Manual serves as a guide by: i) assisting the user with the input of data necessary to solve the boundary value problem, ii.) explaining the relevance of the solution from an engineering standpoint, and iii.) assisting with the visualization of the computer output. An attempt has been made to ascertain and respond to questions most likely to be asked by first time users of SVFLUX.

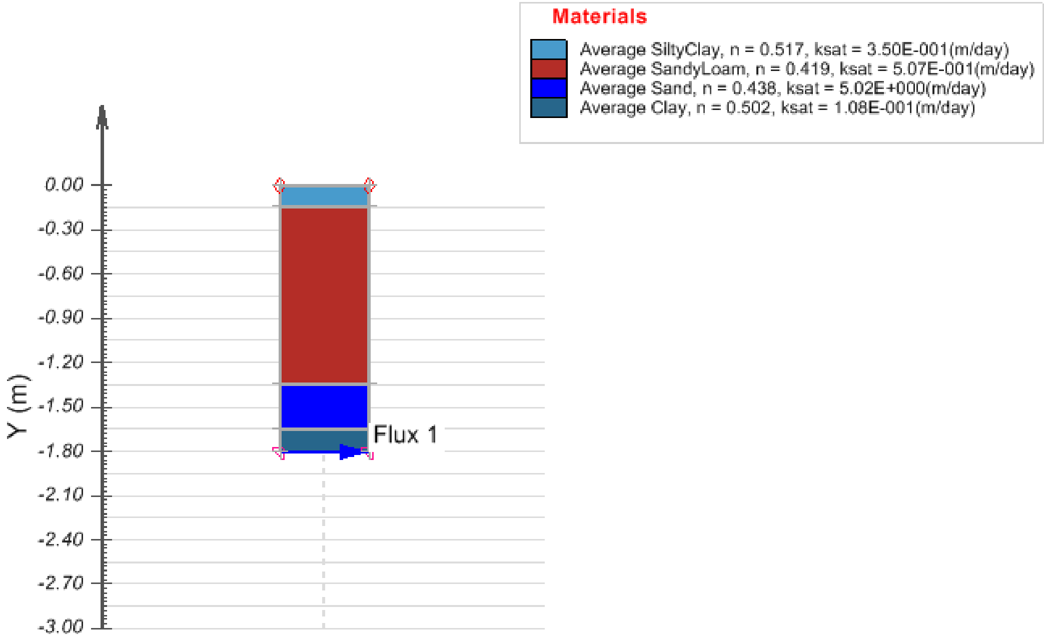
2 1D Cover Design

A simple 1D numerical model has many applications and can provide an initial understanding of a geotechnical problem. The solution is often used as a building block to designing and calibrating more complex multi-dimensional models. This tutorial provides the basic steps for creating a 1D model of an earth cover. The importance of node resolution and proper time increments will be highlighted.

The model will examine the behavior of a 1.8m material column containing 4 distinct materials when subjected to 1 month of climatic events. At the base of the material column is an interim cover overlain by a similar, but higher permeability admixture material. A significant storage layer exists above the admixture with some topsoil as a cover.

Project: EarthCovers
Model: ACAP Helena11
Minimum authorization required: FULL

Model Description and Geometry



2.1 Model Setup

The following steps will be required to set up this model:

a. Create model
b. Enter geometry
c. Specify boundary conditions
d. Apply material properties
e. Specify initial conditions

f. Specify model output
g. Run model
h. Visualize results

a. Create Model

This tutorial assumes that you are already familiar with creating Projects and Models in SVOFFICE. If the "UserTutorial" project does not exist, click "New..." and then enter "UserTutorial" as a new project name.

Since FULL authorization is required for this tutorial, perform the following steps to ensure full authorization is activated:

1. Plug in the USB security key,
2. Go to the *File* > *Authorization* dialog on the SVOFFICE Manager, and
3. Software should display full authorization. If not, it means that the security codes provided by SoilVision Systems at the time of purchase have not yet been entered. Please see the the Authorization section of the SVOFFICE User's Manual for instructions on entering these codes.

To begin modeling this tutorial create a new model in SVFLUX through the following steps:

1. Select a Project under which to organize the tutorial,
2. Press the *New* button under the Models heading,
3. Enter "Cover01" in the Model Name box,
4. Select the following entries:

   Application: SVFLUX
   System: 1D Vertical
   Type: Transient
   Units: Metric
   Time Units: Days (s)
5. Click on the *World Coordinate System* tab,
6. Set the *Y*-Minimum to a value of -2,
7. Set the *Y*-Maximum to a value of 0,
8. Click on the *Time* tab,
9. Enter the following values for time:

   Start Time: 0
   Initial Increment: 0.1
   Maximum Increment: 0.2
   End Time: 30
10. Click the *OK* button to save the model and close the *New Model* dialog,
11. The new model will automatically added to the models list and the new model will be opened, and
12. The *Options* dialog will appear. Select a Vertical Spacing of 0.05 and then click

OK to accept default settings.

b. Enter Geometry (Model > Geometry)

This cover model contains four distinct material layers. The shapes that define each material layer will now be created. The *1D Thicknesses* dialog can be used to quickly create the layer thicknesses:

1. Select *Model > Geometry > 1D Thicknesses*,
2. Enter the following thicknesses in the list box,
   0.15
   1.20
   0.30
   0.15
3. Click *OK*.

c. Specify Boundary Conditions (Model > Boundaries)

Now that all of the regions and the model geometry have been successfully defined, the next step is to specify the boundary conditions. A Unit Gradient boundary will be applied at the base of the column while a Climate boundary condition will be applied to the ground surface.

A climate boundary condition will be applied to the cover model to simulate the rainfall and evaporation for the month of June. The steps for specifying the boundary conditions are thus:

1. Open the *Climate Manager* dialog by selecting *Model > Boundaries > Climate Manager* from the menu,
2. Click the *New* button to open the *New Climate Data* dialog,
3. Enter "June" as the climate dataset name,

- **Define Precipitation**

4. Double-click the precipitation entry for "June" entry to open the *Precipitation Properties* dialog,
5. Check "Include",
6. For Input Option select "Data Linear",
7. Enter the data provided below in the Data list. The data can be cut and pasted from MSExcel here (be sure to select the column headers),
8. Set the Intensity Correction combo box selection to "Global",
9. Switch to the Global Intensity tab,
10. Enter 50% for the Intensity Start,
11. Enter 80% for the Intensity End,
12. Switch to the Runoff tab,
13. Select "Apply" to consider runoff in the solution,
14. Click *OK* to save and close the *Precipitation Properties* dialog.

Time (day)	Flux (m/day)
0	0
2	0.001
8	0.002
13	0.001
23	0.005
28	0.002
29	0.003

- **Define Evaporation**

15. Double-click the evaporation entry for "June" to open the *Evaporation Properties* dialog,
16. Select Include,
17. Select "constant" as the Potential Evaporation method,
18. Move to the *Air Temperature* tab,
19. Enter a constant air temperature of 25^{o}C,
20. Move to the *Air Relative Humidity* tab,
21. Enter a constant relative humidity of 60%,
22. Move to the *Potential Evaporation* tab,
23. Enter a constant potential evaporation of 0.001 m/day,
24. Move to the *Wind Speed* tab,
25. Enter 0 m/day,
26. Click *OK* to save and close the *Evaporation Properties* dialog,
27. Click *OK* to save and close the Climate Manager.

- **Apply boundary conditions to geometry**

1. Select the bottom region by clicking on the region (i.e., or select R4 from the region drop-down list),
2. From the menu select *Model > Boundaries > Boundary Conditions*. The *boundary conditions* dialog will then open. By default the first boundary segment is given a "No BC" value,
3. Select the point (-1.8) in the list,
4. From the Boundary Condition drop down select a Unit Gradient boundary condition,
5. Click *OK*,
6. Select the top region,
7. From the menu select *Model > Boundaries > Boundary Conditions,*
8. Select the point (0.0) in the list,
9. Select a "Climate" boundary condition in the Boundary Condition combo box,
10. Select "June" as the name of the climate object to apply,

11. Click *OK* to both pop-ups that appear,
12. Click *OK*.

d. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the four materials used in the model. This section provides instructions on adding the first material. Repeat the process to add the remaining materials.

1. Open the *Materials* dialog by selecting *Model > Materials > Manager* from the menu,
2. Click the *New...* button to open the *New Materials* dialog,
3. Enter "Silt" for the material name,
4. Set Data Type to Unsaturated,
5. Click *OK* and the *Material Properties* dialog will open,
6. On the *Volumetric Water Content* tab, enter 0.46 as the Saturated VWC,
7. In the SWCC section, select the "Fredlund and Xing Fit",
8. Press the *Properties* button to open the *Fredlund & Xing Fit* dialog,
9. Enter the fit parameters: af = 50.5 kPa, nf = 0.73, mf = 1.8, and hr = 848 kPa,
10. Close the dialog by pressing the *OK* button,
11. Move to the *Hydraulic Conductivity* tab,
12. Enter the ksat value of 8.9E-01 m/day,
13. Select the "Modified Campbell Estimation" under the Unsaturated Hydraulic Conductivity group box,
14. Click the *Properties*... button to open the properties of the Modified Campbell Estimation method,
15. Enter 1e-8 m/day for the k-minimum and 5.0 for the MCampbell p parameter,
16. Click *OK* to save the Modified Campbell parameters,
17. Choose the preferred soil color by clicking on the *Fill Color* button,
18. Click *OK* to save and close the *Material Properties* dialog,
19. Repeat these steps to create the remaining materials. Use the same Modified Campbell Estimation parameters for each of the materials,

I
O
T
E:
To view the Fredlund and Xing SWCC curve fit press the *Graph SWCC* button on the *Fredlund and Xing Fit* dialog.

20. Press *OK* on the *Materials Manager* dialog to close this dialog.

Material Name	USDA Classificatio n	Saturated VWC	Fredlund & Xing af (kPa)	Fredlund & Xing nf	Fredlund & Xing mf	Fredlund & Xing hr (kPa)	ksat (m/ day)

Silt	Silt	0.46	50.5	0.73	1.8	848	0.89
Till	Sandy Loam	0.38	2.9	5.2	0.44	10.6	0.237
Loamy Sand 1	Loamy Sand	0.43	2.3	3.8	0.69	8.4	3.59E-05
Loamy Sand 2	Loamy Sand	0.44	1.8	1.8	1	12	2.4E-05

- **Assign materials to regions**

The next step is to define which materials are applied to which regions.

1. Select *Model > Geometry > Regions*,
2. For each region the appropriate material type must be selected from the combo box. The regions should be numbered from top to bottom (i.e., R1 is the cover material). Click *OK* to any pop-ups which appear. Click *OK* once the assignments have been made. The material assignments should be as follows:

   Silt
   Till
   Loamy Sand 1
   Loamy Sand 2

e. Specify Initial Conditions (Model > Initial Conditions)

Initial conditions must be specified prior to solving a transient seepage model. In this case we will simply specify an initial head which is one meter below the bottom of the model.

1. Select *Model > Initial Conditions > Settings*,
2. Select the "Head Constant/Expression" option,
3. Enter a head of -2.8,
4. Click *OK.*

f. Specify Model Ouput

In this model the plots of interest are the flux at the base of the model and the flux due to the climate events at the top of the model. This section covers how the user may output these types of plots.

There are two methods of seeing output with our software, namely the Plot Manager or the Output Manager. The output for each is described below.

PLOT MANAGER - Plots while the solver is solving

To set up these plots follow the steps below.

1. Open the *Plot Manager* dialog by selecting *Model > Reporting > Plot Manager* from the menu,
2. Click the *Add Defaults*... button,
3. Click *OK* to any pop-ups (* Please note: these defaults may already be present;

if so, you do not need to re-create them),

4. Click *OK*.

The most basic plots have now been defined. As the user becomes familiar with the software additional plots may be created and customized.

OUTPUT MANAGER - Data output for high-quality AcuMesh plots

To set up the output file perform the following steps:

1. Select *Model* > *Reporting* > *Output Manager* from the menu,
2. Click the *Properties* button,
3. Click the *Update Method* tab,
4. Set the time increments to:

   Start: 0

   Increment: 1

   End: 30
5. Click *OK.*

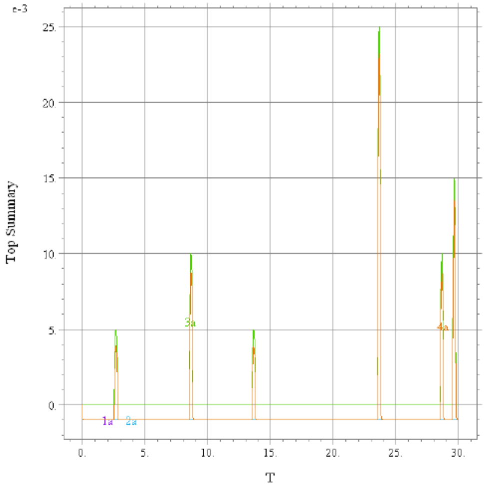
g. Run Model (Solve > Analyze)

The next step is to analyze the model. Select *Solve* > *Analyze* in the menu. This action will write the solver file and open the solver. The solver will automatically begin solving the model.

During solution and after the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the solver. Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots.

The model will now run to solution, but as can be seen in the solver graph "BN1 Top Summary" the precipitation events that were defined are not represented. See the next section for steps on how to provide plot resolution in order to capture climate events.

Now, by observing the "Top Summary" graph in the the solver there are six spikes representing the 6 days with rainfall. The green line shows the precipitation applied to the model while the orange line is the flux across the top boundary. Note that positive flux indicated flow into the model while negative flux is flow (evaporation) out of the model.



Close the solver once the analysis is complete to return to SVFLUX.

h. Visualize Results (Window > AcuMesh)

The visual results for the current model may be examined by selecting the *Window > ACUMESH* menu option.

2.2 Results and Discussions

Preliminary examination of the results can be done by examining the plots produced in FlexPDE. The most commonly viewed type of report is one which summarizes all the cumulative flows in the software. In this case the cumulative plot in FlexPDE will look like the following plot:

Viewing of the presentation plots may be accomplished by the following steps (assuming the user is starting in the front end).

1. Switch to the ACUMESH back end by clicking on the icon on the Process toolbar,
2. Select the *Graphs > Plot Manager* menu option,
3. Select the *Climate* tab, and
4. Click on one of the plots which the user would like to display.

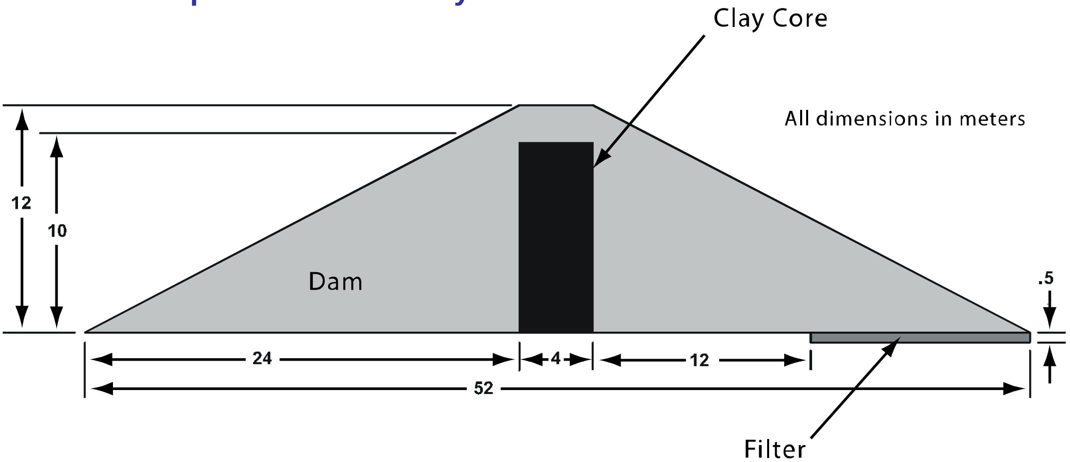
The plots produced may be exported on a variety of formats.

3 2D Earth Dam Example

The following example will introduce some of the features included in SVFLUX and will set up a model of a simple earth fill dam. The purpose of this model is to determine the effects a clay core and filter will have on the final position of the phreatic surface and to determine the total flux that is passing through the dam. The model dimensions and material properties are provided below.

Project: EarthDams
Model: Earth_Fill_Dam
Minimum authorization required: FULL

Model Description and Geometry



Dam X	Dam Y		Core X	Core Y		Filter X	Filter Y
0	0		24	0		40	0
40	0		24	10		40	-0.5
52	0		28	10		52	-0.5
28	12		28	0		52	0
24	12						
20	10						

Material Properties

Silt (Dam): k_s=1.0E-07

Clay (Clay Core): k_s=1.0E-09

Sand (Filter): k_s=1.0E-04

Silt

Suction (kPa)	VWC
0.5886	0.3684
3.306	0.3672

20.01	0.2434
50.03	0.1945
90.06	0.1564
150.1	0.1537

Saturated VWC *n*=0.3672

Boundary Conditions

Dam Region

X	Y	Boundary Condition	Expression
0	0	Zero Flux	
40	0	Continue	
52	0	Continue	
28	12	Continue	
24	12	Continue	
20	10	Head Expression	10

Filter Region

X	Y	Boundary Condition	Expression
40	0	Zero Flux	
40	-0.5	Head Expression	-0.5 m
52	-0.5	Zero Flux	
52	0	Continue	

3.1 Model Setup

To set up the model described in the preceding section, the following steps will be required:

a. Create model
b. Enter geometry
c. Specify initial conditions
d. Specify boundary conditions
e. Apply material properties
f. Specify model output
g. Run model
h. Visualize results

These outlined steps are detailed in the following sections.

a. Create Model

Since FULL authorization is required for this tutorial, perform the following steps to ensure full authorization is activated:

1. Plug in the USB security key,
2. Go to the *File* > *Authorization* dialog on the SVOFFICE Manager, and
3. Software should display full authorization. If not, it means that the security codes provided by SoilVision Systems at the time of purchase have not yet

been entered. Please see the the Authorization section of the SVOFFICE User's Manual for instructions on entering these codes.

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,
2. Select "ALL" under the Applications combo box and "ALL" for the Model Origin combo box,
3. Create a new project called UserTutorial by pressing the *New* button next to the list of projects,
4. Create a new model called Earth_Fill_Dam_User by pressing the *New* button next to the list of models. The new model will be automatically added under the recently created UserTutorial project. Use the settings below when creating this new model,
5. Select the following:

Application:	SVFLUX
System:	2D
Type:	Steady-State
Units:	Metric
Time Units:	Seconds (s)

6. Access the View menu and select World Coordinate System,
7. Enter the World Coordinates System coordinates shown below into the dialog,

x - minimum:	-5
y - minimum:	-5
x - maximum:	55
y - maximum:	40

8. Click *OK* to close the dialog.

The workspace grid spacing needs to be set to aid in defining region shapes. The filter portion of the model has coordinates of a precision of 0.5m. In order to effectively draw geometry with this precision using the mouse, the grid spacing must be set to a maximum of 0.5.

1. Select *View* > *Options* from the menu,
2. Enter 0.5 for both the horizontal and vertical spacing,
3. Click *OK* to close the dialog.

b. Enter Geometry (Model > Geometry)

Model geometry is defined as a series of layers and can be either drawn by the user or defined as a set of coordinates. Model Geometry can be imported from either .DXF files or from existing models.

This model will be divided into three regions, which are named Dam, Core, and Filter. Each

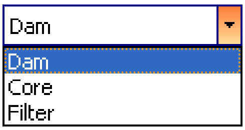
region will have one of the materials specified as its material properties. To add the necessary regions follow these steps:

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions* from the menu,
2. Change the first region name from R1 to Dam. To do this, highlight the name and type new text,
3. Press the *New* button to add a second region and name it Core,
4. Click *New* to add the third material and name it Filter,
5. Click *OK* to close the dialog.

The shapes that define each material region will now be created. Note that when drawing a geometric shape, information will be added to the region that is current in the Region Selector. The Region Selector is at the top of the workspace, in the toolbar.

- **Define the Dam**

1. Ensure the Dam region is current in the region selector,



2. Ensure that the SNAP and GRID options in the status bar are set to ON,
3. Select *Draw > Model Geometry > Region Polygon* from the menu,
4. The cursor will now be changed to cross hairs,
5. Move the cursor near (0,0) in the drawing space. You can view the coordinates of the current position the mouse is at in the status bar just above the command line,
6. To select the point as part of the shape left click on the point,
7. Now move the cursor near (40,0) and then left click on the point. A line is now drawn from (0,0) to (40,0),
8. Now move the cursor near (52,0) and then left click on the point. A line is now drawn from (40,0) to (52,0),
9. Move the cursor near the point (28,12) and left click on the point and a line is now drawn from (52,0) to (28,12),
10. Move the cursor near the point (24,12). Left click on the point to draw a line from (28,12) to (24,12),
11. Move the cursor near the point (20,10). Double click on the point to finish the shape. A line is now drawn from (24,12) to (20,10) and the shape is automatically finished by SVFLUX by drawing a line from (20,10) back to the start point, (0,0).

Note: in this example regions are entered in a counter-clockwise order. Generally speaking,

the order in which region node points are entered (clockwise/counter-clockwise) is not significant.

Repeat this process to define the Core and Filter regions according to the data provided at the start of this tutorial. The geometry for the model may be obtained in the spreadsheet located here.

NOTE:
You may find it easier to enter your coordinates from the *Region Properties* dialog.

If an error is made when entering the region geometry the user may recover from the error and start again by one of the following methods:
a. Press the escape (esc) key
b. Select a region shape and press the *delete* key
c. Use the Undo function on the Edit menu

If all model geometry has been entered correctly the shape should look like the diagram at the beginning of this tutorial.

c. Specify Initial Conditions (Model > Initial Conditions)

Initial conditions are generally associated with transient model runs. Their purpose is to provide a reasonable starting point for the solver. In a steady-state model, initial conditions can be used to "precondition" the solver to allow faster convergence. However, this "preconditioning" is unnecessary for the current model. Therefore, the initial conditions should be set to "None" using the following steps:

1. Select *Model > Initial Conditions > Settings*,
2. Select the "None" option,
3. Press *OK* to close the dialog.

d. Specify Boundary Conditions (Model > Boundaries)

Boundary conditions must be applied to region points. Once a boundary condition is applied to a boundary point the starting point is defined for that particular boundary condition. The boundary condition will then extend over subsequent line segments around the edge of the region in the direction in which the region shape was originally entered. Boundary conditions remain in effect around a shape until re-defined. The user may not define two different boundary conditions over the same line segment.

More information on boundary conditions can be found in *Menu System > Model Menu > Boundary Conditions > 2D Boundary Conditions* in your User's Manual.

Now that all of the regions and the model geometry have been successfully defined, the next step is to specify the boundary conditions. A head of 10 m will be defined on the upstream face of the Dam region with the Zero Flux condition being applied to the remainder. The Core will be set to a Zero Flux condition by default and will not need to be altered. The Filter will have a head of –0.5m at its base. The steps for specifying the boundary conditions are as follows:

- **Dam**
  1. Select the "Dam" region in the region selector,
  2. From the menu select *Model > Boundaries > Boundary Conditions*. The *boundary conditions* dialog will open. By default the first boundary segment will be given a No BC condition,
  3. Select the point (20,10) from the list,

4. From the Boundary Condition drop-down select a Head Expression boundary condition. This will cause the Expression box to be enabled,
5. In the Constant/Expression box enter a head of 10,
6. Click *OK* to save the input Boundary Conditions and return to the workspace,

NOTE:

The Zero Flux boundary condition for the point (0,0) becomes the boundary condition for the following line segments that have a Continue boundary condition until a new boundary condition is specified.

In this case the line segments from (0,0) to (52,0), (52,0) to (28,12), (28,12) to (24,12), and (24,12) to (20,10) are all given a zero flux boundary condition. By specifying a head of 10 at point (20,10) the boundary condition for the line segment extending from (20,10) to (0,0) is set to a head equal to 10m.

- **Filter**

7. Select the "Filter" region in the region selector,
8. From the menu select *Model* > *Boundaries* > *Boundary Conditions* to open the *Boundaries* dialog,
9. Select the point (40,-0.5) from the list,
10. In the Boundary Condition drop-down select a Head Expression boundary condition,
11. Enter the value -0.5 in the Constant/Expression box,
12. Select the point (52,-0.5) from the list,
13. In the Boundary Condition drop-down select a Zero Flux boundary condition,
14. Click *OK* to save the input Boundary Conditions and return to the workspace.

e. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the three materials that will be used in the model. A clay is defined for the core, the silt will make up the dam, and the filter will consist of a sand. This section will provide instructions on creating the clay material. Repeat the process to add the other two materials.

SILT - Soil-water characteristic curve

SWCC material properties for the Silt material must be set. These values can be found at the beginning of this tutorial model. Follow these steps to set up the SWCC values for the Silt material.

1. Open the *Materials* dialog by selecting *Model* > *Materials* > *Manager* from the menu,
2. Click the *New*... button to create a material and enter the name "DamSilt",
3. Set Data Type to Unsaturated,
4. Select the *Volumetric Water Content* tab,
5. Enter the Saturated VWC value 0.3672,
6. In the SWCC area, select "Fredlund & Xing Fit",
7. Press the *Data* button,

8. Enter the table of values found at the beginning of this tutorial model,
9. Once all of the values have been entered, press *OK* to accept the changes,
10. Click the *Properties* button to open the *Fredlund & Xing fit* dialog,
11. Click the *Apply Fit* button to have the material parameters estimated by the Fredlund & Xing method (using the data entered above),
12. Click the *OK* button to accept the entered information.

SILT - Hydraulic conductivity

1. Click the Hydraulic Conductivity tab,

> **NOTE:**
> When a new material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned to it will display that material's fill color.

2. Enter the ksat value of 1.000E-07 m/s,
3. Select Modified Campbell in the Unsaturated Hydraulic Conductivity drop-down,
4. Click the *Properties...* button beside the Modified Campbell combo box,
5. Enter a k minimum value of 1e-9 m/s,
6. Enter a Modified Campbell 'p' value of 5,
7. Press *OK* twice to close the dialogs,

FILTER SAND - Soil-water characteristic curve

1. Click the *New*... button to create a material,
2. Enter "Filter Sand" into the name text field,
3. Set Data Type to Unsaturated,
4. Select the *Volumetric Water Content* tab,
5. Enter a Saturated VWC value of 0.35 (this input is not theoretically required for solving a steady-state model but does enable additional plotting facilities),
6. Leave the SWCC combo box at a setting of None.

FILTER SAND - Hydraulic conductivity

1. Click the Hydraulic Conductivity tab,
2. Enter a ksat value of 0.0001 m/s,
3. Leave the Unsaturated Hydraulic Conductivity setting at None,
4. Click *OK* to close the *Material Properties* dialog.

CORE CLAY - Soil-water characteristic curve

1. Click the *New*... button to create a material,
2. Enter "Core Clay" into the name text field,

3. Set Data Type to Unsaturated,
4. Select the *Volumetric Water Content* tab,
5. Enter a Saturated VWC value of 0.40 (this input is not theoretically required for solving a steady-state model but does enable additional plotting facilities),
6. Leave the SWCC combo box at a setting of None,

CORE CLAY - Hydraulic conductivity

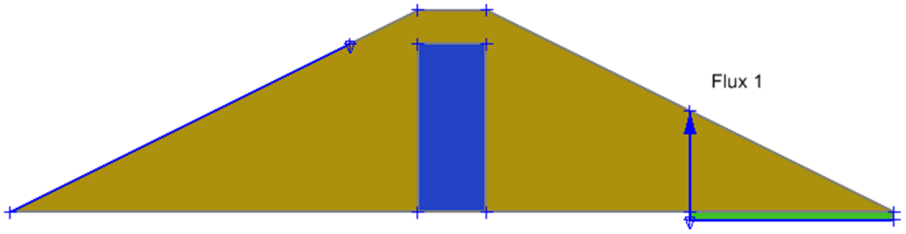
1. Click the Hydraulic Conductivity tab,
2. Enter a ksat value of 1E-9 m/s,
3. Leave the Unsaturated Hydraulic Conductivity setting at None,
4. Click *OK* to close the *Material Properties* dialog,
5. Click *OK* to close the Material Manager.

Once all material properties have been entered, we must apply the materials to the corresponding regions.

1. Open the *Region Properties* dialog by selecting *Model* > *Geometry* > *Region Properties* from the menu,
2. Select the "Dam" region by using the arrows at the top right of the dialog and assign the Silt material to this region,
3. Select the "Core" region and assign the Core Clay material to this region,
4. Select the "Filter" region and assign the Filter Sand material to this region,
5. Press the *OK* button to accept the changes and close the dialog.

f. Specify Model Output

Flux sections are used to report the rate of flow across a portion of the model for a steady state analysis, and the rate and volume of flow moving across a portion of the model in a transient analysis. For the current model a flux section will be created at the location shown below.



1. Select *Draw* > *Flux Section* from the menu,
2. Move the cursor to the point (40,0) and click on it,
3. To finish the Flux Section, move the cursor to the point (40,6) and double-click on it. A blue line with an arrow on the end should be drawn across the dam,
4. Select *Model* > *Reporting* > *Flux Sections* from the menu. The *Flux Section List*

dialog will open,

5. Select Flux 1 from the list,
6. Click "Properties" to open the *Flux Section Properties* dialog. This dialog displays the properties of the current flux section which the user can update manually as well as graphically. Press the *OK* buttons on the remaining dialogs,
7. Open the *Plot Properties* dialog by selecting *Model > Reporting > Plot Manager*. Select the *Flux Sections* tab. All available flux section reports will be presented in the list box. Several methods (*x, y,* normal) of reporting the flux will be present. Click on the uppermost report (Variable is "Flux") in the list box, and click "Properties". Click "Dam" from the Restrict to Region drop-down,
8. Press *OK* to close the dialog,
9. Press OK to close the *Plot Manager* dialog,
10. Notice that the flux section label, 'Flux 1', is partially on the region boundary in the workspace. To move the label location, select the textbox in the workspace and drag it to the desired location.

NOTE:
Flux Section labels can be formatted in the same manner as regular textboxes.

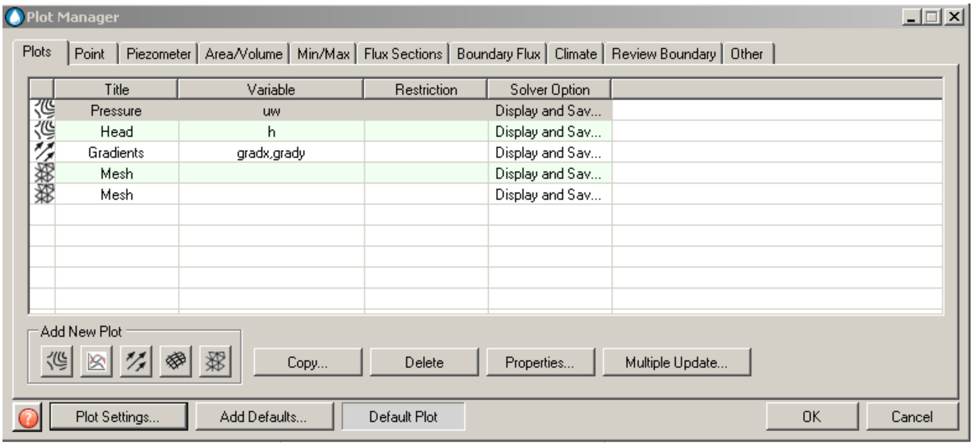
Two levels of output may be specified: i) output (graphs, contour plots, fluxes, etc.) which are displayed during model solution, and ii) output which is written to a standard finite element file for viewing with ACUMESH software. Output is specified in the following two dialogs in the software:

i) Plot Manager:	Output displayed during model solution.
ii) Output Manager:	Standard finite element files written out for visualization in ACUMESH or for initial condition input to other finite element packages.

PLOT MANAGER (Model > Reporting > Plot Manager)

The plot manager dialog is first opened to display appropriate solver graphs. There are many plot types that can be specified to visualize the results of the model. A few will be generated for this tutorial example model, including a plot of the solution mesh, pressure contours, head contours, and gradient vectors.

1. Open the *Plot Manager* dialog by selecting *Model > Reporting > Plot Manager* from the menu,



2. The toolbar at the bottom left of the dialog contains a button for each plot type. Click on the *Contour* button to begin adding the first contour plot. The *Plot Properties* dialog will open,
3. Enter the title 'Pressure',
4. Select 'uw' as the variable to plot from the drop-down,
5. Click *OK* to close the dialog and add the plot to the list,
6. Repeat steps 2 to 5 to create the plots shown in the above dialog,
7. Click *OK* to close the *Plot Manager* and return to the workspace.

Alternatively, the user may press the *Add Default Plots* button and typical plots will be added to the plot list.

g. Run Model (Solve > Analyze)

The next step is to analyze the model. Select *Solve* > *Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model. After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the FlexPDE solver. Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots. This section will give a brief analysis for each plot that was generated.

These reports are intended to provide the user with low-quality graphs which give a rough indication of the results. Creating professional-quality visualizations of the results can be accomplished with ACUMESH software, the use of which is described below.

The Flux through the model is displayed in the dialog of report showing a breakdown of the *X, Y,* and Normal components of flow through the model.

FlexPDE will report the Flow information. For this model it is as follows:

X Component of Flow in (m^3/s)=	2.49 e-8
Y Component of Flow in (m^3/s)=	0.00
Normal Flow in (m^3/s)=	2.49 e-8

h. Visualize Results (Window > AcuMesh)

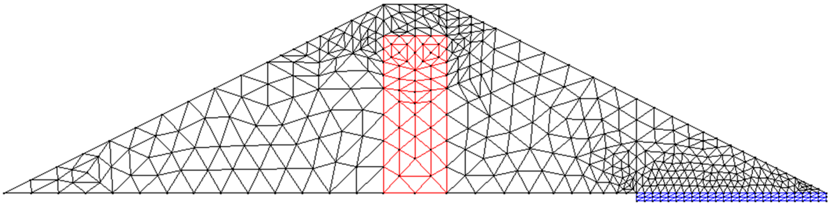
The visual results for the current model may be examined by selecting the *Window > ACUMESH* menu option.

3.2 Results and Discussions

The following plots are typically desired for a seepage analysis. Each plot, as well as a brief description, is displayed below.

- **Solution Mesh**

Once you have analyzed the model, the default display in ACUMESH displays the finite element mesh used to obtain the solution.

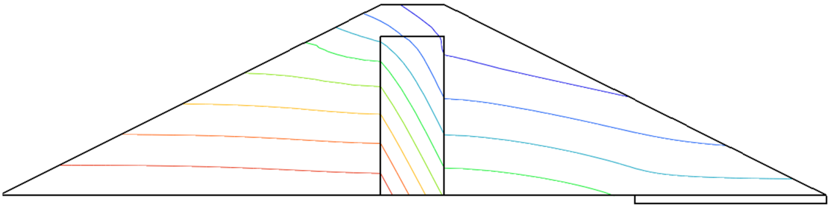


The Mesh plot displays the finite-element mesh generated by the solver. The mesh is automatically refined in critical areas such as the Dam – Filter contact where there is a significant change in hydraulic conductivity.

The uncolored mesh should now be displayed in ACUMESH. In order to display contours of pressure head the user must:

1. Select *Plot > Contours* from the menu,
2. Under the Regions tab, click "Show All" button,
3. Select 'uw' as the Variable Name,
4. Press *OK* to close the dialog.

- **Pressure Contours**



The most important contour in the above plot is the one that corresponds to zero pressure. This contour represents the phreatic surface. All material that lies below this line is saturated and all material that lies above this line is considered to be unsaturated. The above design would be acceptable as the water table exits the dam at the beginning of the filter. If the water table had extended to the toe of the dam, there would be concern that the toe of the dam would become unstable due to piping failure.

The mesh can be turned off for certain regions through the following process:

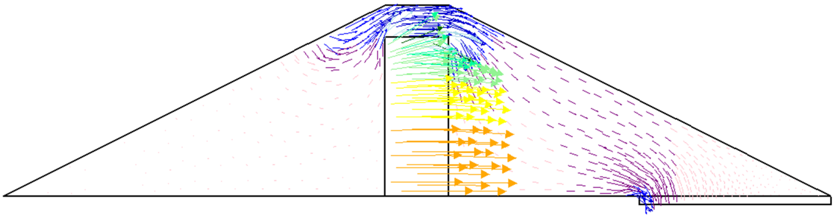
1. Select *Plot > Mesh* from the menu,

2. Select 'Core' in the list and uncheck the checkbox,
3. Press *OK* to close the dialog.

- **Flow Vectors**

Flow vectors can be displayed through the following process:

1. Select *Plot* > *Vectors* from the menu,
2. Select "Dam" in the list and ensure the checkbox is checked,
3. Press *OK* to close the dialog.



Flow Vectors show both the direction and the magnitude of the flow at specific points in the model. The low conductivity of the core causes the majority of the flow to go up and over the core causing increased gradients in this area. The other area of interest is at the filter. Vectors illustrate that flow is exiting the dam in this region.

- **Head Contours**

As expected, most of the head is dissipated in the core of the dam. This is illustrated in by how close the contours are in the core. The maximum head in the model occurs on the upstream face of the dam and is equal to ten. This is expected, as this was the boundary condition set on the upstream face of the dam. The lowest head occurs at the filter and is equal to –0.5m.

The current visualization can be exported on a standardized format through the following steps:

1. Select *File* > *Export As* from the menu,
2. Specify a file name, and
3. Specify a file type.

4 2D Stochastic Example

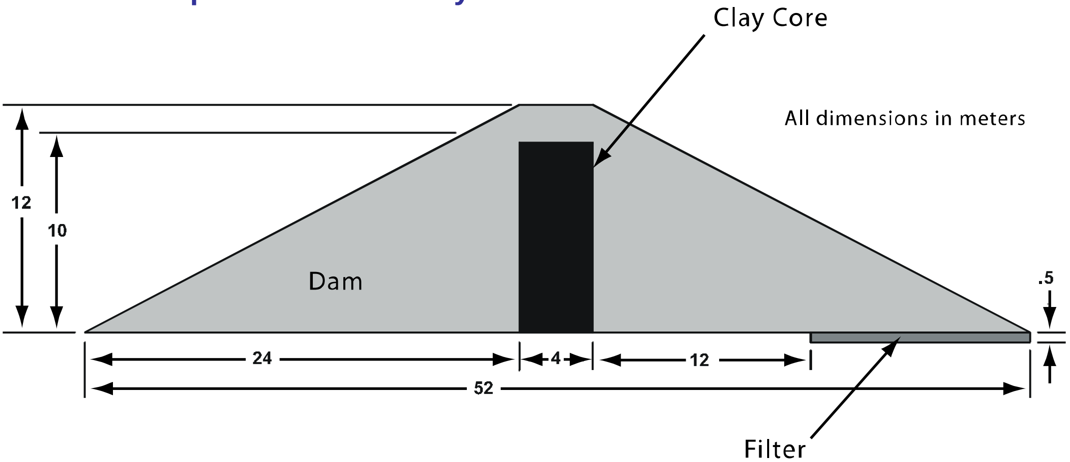
A single run of a numerical model is often not sufficient. SVFLUX implements the ability to incorporate statistical uncertainty into the input of material properties. This tutorial section guides the user through the setup and execution of a model implementing the stochastic or statistical features available in SVFLUX.

The earth dam example presented in Section 2 is extended to incorporate a variation in the slope of the unsaturated portion of the hydraulic conductivity curve. The current example makes use of the Modified Campbell method of estimating the unsaturated hydraulic conductivity. The slope of the unsaturated portion is controlled by a "*p*" parameter. Instead of specifying a single value for this parameter a Monte Carlo analysis will be used to generate 50 runs of the same model varying the "*p*" parameter of the Modified Campbell estimation method.

Why do we want to do this? In this example we want to determine the influence of the slope of the unsaturated hydraulic conductivity curve on the difference in flow through the core or over the top of core. The unsaturated hydraulic conductivity curve is often not measured and it is useful to determine the sensitivity of flow through various portions of the dam on this curve.

Project: EarthDams
Model: EarthFillDamSto100
Minimum authorization required: FULL

Model Description and Geometry



4.1 Model Setup

The following steps will be required to set up this model:

a. Create model
b. Specify sensitivity material properties
c. Specify model output
d. Run model
e. Visualize results

a. Create Model

Since FULL authorization is required for this tutorial, perform the following steps to ensure full authorization is activated:

1. Plug in the USB security key,
2. Go to the *File > Authorization* dialog on the SVOFFICE Manager,
3. Software should display full authorization. If not, it means that the security codes provided by SoilVision Systems at the time of purchase have not yet been entered. Please see the the Authorization section of the SVOFFICE User's Manual for instructions on entering these codes.

In order to create the stochastic model, save a copy of the 2D example model created in Section 2. This is accomplished through the following steps:

1. Select the "UserTutorial" project and Open the Earth_Fill_Dam_User model,
2. Select *File > Save As*,
3. Type the name User_EarthFillDamSto and click *OK*.

Now a new model has been created and loaded into the workspace that will be modified to include stochastic analysis.

The second step is to describe the number of stages/runs that will be used in the analysis. A series of 50 model runs will be performed using the Monte Carlo analysis technique. The user must first set the number of model stages to 50 by following these steps:

1. Open the *Model > FEM Options* dialog,
2. Open the Advanced section of the *FEM Options* dialog,
3. STAGES field should be set to 50 under the Stage Controls area of the dialog,
4. Click *OK* to close the dialog and accept the changes.

b. Specify Sensitivity Material Properties (Model > Materials)

The next step that must occur is that the Modified Campbell "*p*" parameter will be staged. In this example the parameter will be set to have a mean value of 5 and a standard deviation of 2. This staging is accomplished through the following process:

1. Select *Model > Materials > Manager* to open the list of current materials,
2. Select the "DamSilt" material and click the *Properties* button,
3. Click the *Stage Parameters* button located in the lower left of the dialog,
4. Select the "Mcampbell p parameter" and click the *Include* button,
5. *Click the Stage Values* button*,*
6. Click the *Stage Distribution* button,
7. Select "Monte Carlo Normal" as the distribution method,
8. Enter a mean of 5 and a standard deviation of 2 and click the *OK* button,
9. A list of 50 generated values should now be displayed in the list box,
10. Press *OK* on all remaining dialogs to close them.

The parameter has now been staged.

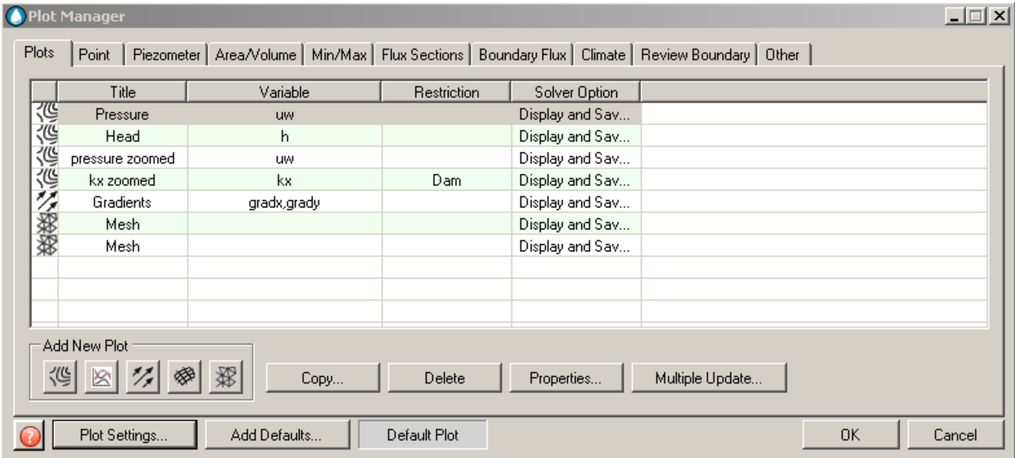
c. Specify Model Output

Two levels of output may be specified: i) output (graphs, contour plots, fluxes, etc.) which are displayed during model solution, and ii) output which is written to a standard finite element file for viewing with ACUMESH software. Output is specified in the following two dialogs in the software:

i) Plot Manager: Output displayed during model solution.
ii) Output Manager: Standard finite element files written out for visualization in ACUMESH or for inputting to other finite element packages.

PLOT MANAGER (Model > Reporting > Plot Manager)

The *Plot Manager* dialog is first opened to display appropriate solver graphs. There are many plot types that can be specified to visualize the results of the model. The user may press the *Add Default Plots* button to add typical plots. Define the plots shown in the below screenshot.



Specify the following Zoom settings for the pressure zoomed contour plot:

X: 24.73
Y: 8.974
width: 4.936
height: 4.026

Specify the following Zoom settings for the kx zoomed contour plot:

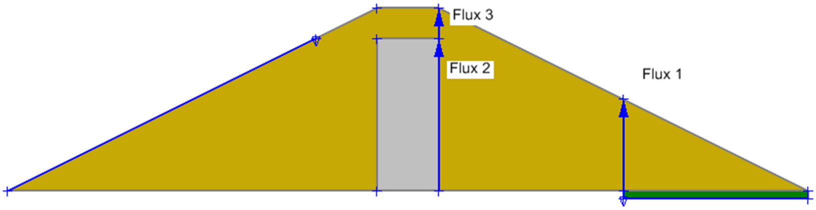
X: 25.06
Y: 9.211
width: 4.278
height: 3.671

OUTPUT MANAGER (Model > Reporting > Output Manager)

There are four output file types that can be specified to export the results of the model. Two will be generated for this tutorial example model: a transfer file of heads, and a plot to transfer the results to ACUMESH (created by default).

1. Open the *Output Manager* dialog by selecting *Model > Reporting > Output Manager* from the menu,
2. The toolbar at the bottom left corner of the dialog contains a button for each output file type. Click on the *SVFlux* button to begin adding the output file. The *Output File Properties* dialog will open,
3. The default settings for this output file are good for this model. Click *OK* to close the dialog and add the output file to the list,
4. Click *OK* to close the *Output Manager* and return to the workspace.

Note that these settings should already be present if the user has copied this example from the previous tutorial. Since we are trying to determine the amount of flow through different portions of the dam we will place flux sections at the right hand side of the dam core in addition to the flux section already present near the filter.



The flux sections should have the following coordinates: Flux 2 (28,0) to (28,10), Flux 3 (28,10) to (28,12).

The flux sections may be drawn graphically or entered through the *Model > Reporting > Flux Sections* menu option. Refer to Specifying a Flux Section in this manual for instructions on adding Flux Sections.

Each flux section must have their flows restricted to the Dam region. In order to do this, complete the following steps:

1. Select *Model > Reporting > Plot Manager* from the menu,
2. Select the *Flux Sections* tab,
3. Select the Flux 1 report plot and click Properties,
4. Select the "Dam" region in the Restrict to Region drop-down,
5. Repeat steps 3 and 4 for all Flux report plots,
6. Click *OK* to close the *Plot Manager* and accept the above changes.

d. Run Model (Solve > Analyze)

The next step is to analyze the model. Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model.

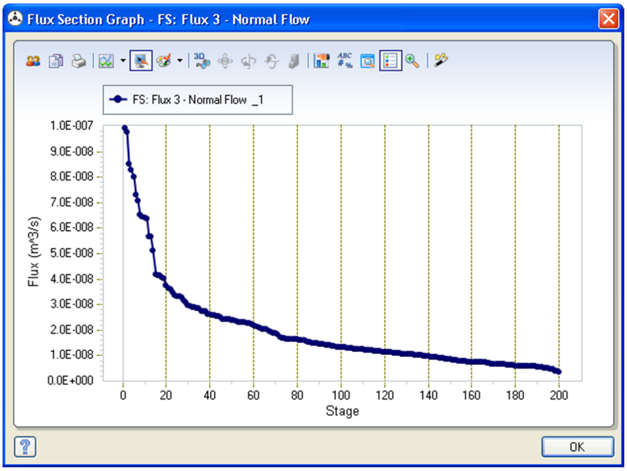
e. Visualize Results (Window > AcuMesh)

The visual results for the current model may be examined by selecting the *Window > ACUMESH* menu option. The user may also view the results by pressing the ACUMESH icon on the process toolbar.

4.2 Results and Discussions

Selecting *Solve > Analyze* from the menu will initiate model solution. Attention should be directed to the stage display in the finite element solver. Once a particular stage is complete the next stage will automatically be initiated. The various flows through differing portions of the earth dam may be seen in the output of the flux section history plots for flux sections 2 and 3.

From these history plots it can be seen that the model is quite sensitive to the slope of the unsaturated hydraulic conductivity curve.



In order to view the history graph on the Flux 3 Flux section in ACUMESH, follow these instructions:

1. Select the *Plot Manager Graphs* dialog by selecting *Graphs > Plot Manager*,
2. Once this dialog opens, select the *Flux Section* tab and choose the Flux 3 history plot, and
3. The graph shown above should now appear. You can format the axis display settings by choosing the *Properties* button, . Options on the scale, labels, and gridlines are available for users to change the graph display to their liking.

5 3D Reservoir

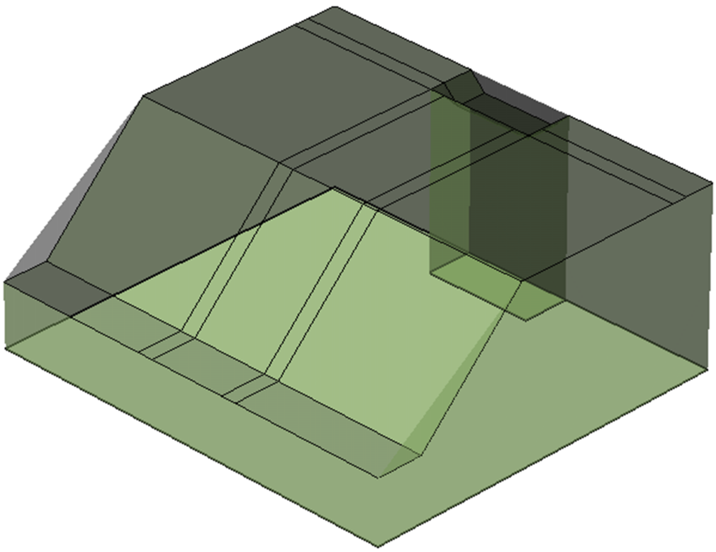
The following example will introduce you to the three-dimensional SVFLUX modeling environment. This example is used to investigate the flow and pressure conditions existing within a slope due to a holding pond at the crest. The example is modeled using two regions, two surfaces, and one material. The model data and material properties are provided below.

The purpose of this model is to determine the flow out of the reservoir.

Project: Ponds
Model: Reservoir3D
Minimum authorization required: STUDENT

Model Description and Geometry

A small holding pond of dimensions 27m wide and 24m long is created at the top of the slope. A slope of angle 45° begins 10m from the edge of the holding pond. The slope has a total height of 11m and levels off for a distance of 3m.



Material Properties

Tutorial Material - Reservoir3D: k_{sat} = 2.512E-06 kPa

5.1 Model Setup

The following steps will be required to set up the model:

a. Create model
b. Enter geometry
c. Specify initial conditions
d. Specify boundary conditions
e. Apply material properties
f. Specify model output
g. Run model

h. Visualize results

a. Create Model

The first step in defining a model is to decide the project under which the model is going to be organized. If the project is not yet included you must add the project before proceeding with the model. In this case, the model is placed under a project called "UserTutorial".

In order to add this project follow these steps:

1. Access the *SVOFFICE Manager* dialog. If a project named Tutorial already exists, skip to the next section,
2. Click *New* in the upper right of the Projects section,
3. The *Create New Project* dialog is opened along with a prompt asking for a new Project Name,
4. Type "UserTutorial" as the new Project Name and press *OK,*

The *Project Properties* dialog is where information specific to each project is stored. This will include the Project Name, Project Folder, and Project Notes information.

> **NOTE:**
> The Project Name is the only required information needed to define a project. The rest of the fields are optional.

The dialog is opened ready to accept information. It should be noted that once the project is defined it will be identified by the Project Name throughout the rest of the program. Also, SVFLUX does not allow you to specify two projects with the same Project Name.

5. Fill out the dialog with the desired information,
6. To exit this dialog and return to the *SVOFFICE Manager*, click *OK*. The project information is automatically saved upon entry.

If the project is not yet included, you must add the project before proceeding with the model, as detailed in the previous section.

When the *SVOFFICE Manager* dialog is opened there will be a list of the projects that have been defined. To add a model to the UserTutorial project follow these steps:

1. Select the "UserTutorial" project from the Project Name list,
2. Press the *New* button under the 'Models' heading,
3. Enter User_Reservoir_3D in the Model Name box*,*
4. Select the following:

Application:	SVFLUX
System:	3D
Type:	Steady-State
Units:	Metric
Time Units:	Seconds (s)

World Coordinate System

xmin = -5 ymin = -5 zmin = -5

xmax = 30 ymax = 30 zmax = 20

5. Click the *OK* button to save the model and close the *New Model* dialog,
6. The new model will automatically be opened in the workspace.

The workspace grid spacing needs to be set to aid in defining region shapes. The geometry data for this model has coordinates of a precision of 1m. In order to effectively draw geometry with this precision using the mouse, the grid spacing must be set to a maximum of 1.

1. Select *View* > *Options* from the menu,
2. Enter 1 for both the horizontal and vertical spacing,
3. Click *OK* to close the dialog.

b. Enter Geometry (Model > Geometry)

A region in SVFLUX is the basic building block for a model. A region represents both a physical portion of material being modeled and a visualization area in the SVFLUX CAD workspace. A region will have a set of geometric shapes that define its material boundaries. Also, other modeling objects including features, flux sections, water tables, text, and line art can be defined on any given region.

This model will be divided into two regions, which are named Slope and Reservoir. To add the necessary regions follow these steps:

1. Open the *Regions* dialog by selecting *Model* > *Geometry* > *Regions* from the menu,
2. Change the first region name from R1 to Slope. To do this, highlight the name and type the new text,
3. Press the *New* button to add a second region,
4. Change the name of the second region to Reservoir,
5. Click *OK* to close the dialog.

The shapes that define each region will now be created. Note that when drawing geometry shapes, the region that is current in the Region Selector is the region the geometry will be added to. The Region Selector is at the top of the workspace. Refer to the 3D Example Model Data section for the geometry points for each region.

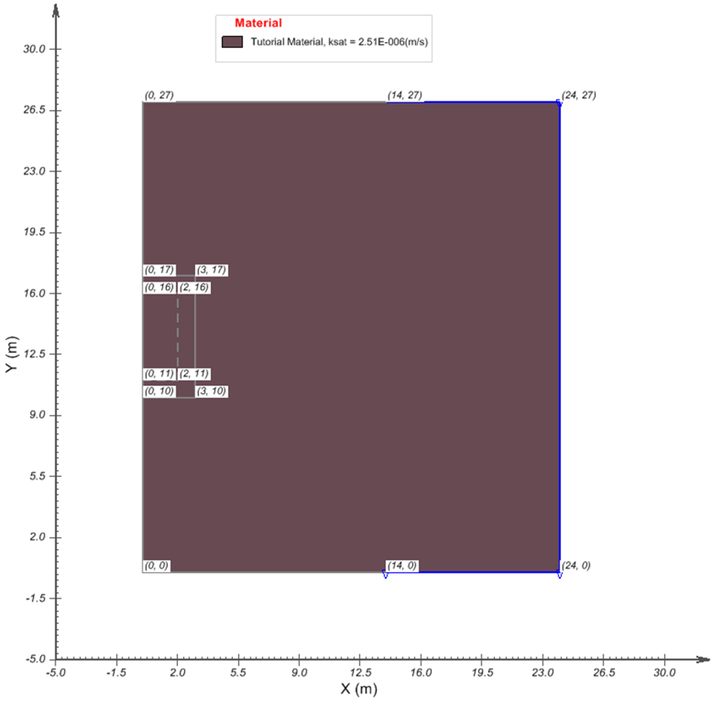
- **Define the Slope region**

1. Select "Slope" in the Region Selector, found in the toolbar at the top of the workspace,
2. Select *Draw* > *Model Geometry* > *Region Polygon* from the menu,
3. Move the cursor near (0,0) in the drawing space. You can view the coordinates of the mouse's current position in the status bar just below the drawing space. The SNAP and GRID options in the status bar must both be On; OSNAP should

be Off,

4. To select the point as part of the shape left click on the point,
5. Now move the cursor near (14,0) and left click on the point. A line is now drawn from (0,0) to (14,0),
6. Repeat the process for the following coordinates: (24,0), (24,27), and (14,27),
7. For the final point, (0,27), double-click on the point to finish the shape. The shape is automatically finished by SVFLUX by drawing a line from (0,27) back to the start point, (0,0),

If the Main geometry has been entered correctly, the shape should be similar to the following:



NOTE:
If a mistake was made entering the coordinate points for a shape, edit the shape using the *Region Properties* dialog (menu item *Model > Geometry > Region Properties*).

- **Define the Reservoir**

8. Select "Reservoir" in the Region Selector,
9. Select *Draw > Model Geometry > Region Polygon* from the menu,
10. Move the cursor near (0,10) in the drawing space. You can view the coordinates of the current position the mouse is at in the status bar just below the drawing space,

11. When the cursor is near the point, click the left mouse button. The cursor should automatically snap to the point (0,10) as long as the cursor is close to the point, the SNAP and GRID options in the status bar are both on, and the "OSNAP" option is off,
12. Now move the cursor near (3,10). Left click on the point. A line is now drawn from (0,10) to (3,10),
13. Repeat this process for the coordinate (3,17). If a mistake is made simply press the *Delete* key on the keyboard and start over,
14. For the final point, (0,17), double-click on the point to finish the shape. A line is now drawn from (3,17) to (0,17) and the shape is automatically finished by SVFLUX by drawing a line from (0,17) back to the start point, (0,10).

NOTE:
At times it may be tricky to snap to a grid point that is near a line defined for a region. Turn the object snap off by clicking on "OSNAP" in the status bar to alleviate this problem.

This model consists of two surfaces and each will be defined by a different method. By default every model initially has two surfaces.

- **Define Surface 1**
  This surface will be defined by providing a constant elevation.

1. Select "Surface 1" in the Surface Selector found at the top of the workspace,
2. Select *Model > Geometry > Surface Properties* from the menu to open the *Surface Properties* dialog,
3. For the Surface Definition Option, select Expression from the drop-down,
4. Click on the Expression tab,
5. Enter a Surface Expression of 0,
6. Click *OK* to close the dialog,

- **Define Surface 2**

This surface will be defined by providing a grid of (*X,Y*) points and corresponding elevations.

7. Select "Surface 2" in the Surface Selector,
8. Go to *Model > Geometry > Surface Properties* in the menu to open the *Surface Properties* dialog,
9. Select "Elevation Data" from the Definition Options drop-down and click the *Define Gridlines* button to set up the grid for the selected surface,
10. Click the *Add Irregular* button to open the *Add Irregular X Gridlines* dialog,
11. Enter 0, 2, 3, 14, 21, and 24,
12. Click *OK* to add the gridlines and close the dialog,
13. Select 10 from the list,
14. Press *Delete*; then click "Yes" to confirm the deletion,

15. Move to the *Y Grid Lines* tab and click the *Add Irregular* button to open the *Add Irregular Y Gridlines* dialog,
16. Enter 0, 10, 11, 16, 17, and 27,
17. Click *OK* to add the gridlines and close the dialog,
18. Press *OK* to close the *Define Grid* dialog. Leave the *Surface Properties* dialog open,

NOTE:
You can import *XYZ* data, paste surface data, or paste surface grid data in this menu for faster data entry.

Now that the grid has been set up, elevations must be specified for all the grid points:

19. Select "Elevation Data" from the Surface Definition Options drop-down in the *Surface Properties* dialog,
20. Enter the missing Z elevations as provided in the 3D Example Model Data section at the end of this tutorial,
21. Press *OK* to close the dialog,

The grid for surface 2 can be made viewable in the workspace.

22. Select *Model* > *Geometry* > *Surfaces* from the menu,
23. Click the Grid check box for Surface 2. A check mark should appear in the box,
24. Click the Grid check box for Surface 1. The check mark should disappear,
25. Press *OK* to close the dialog.

c. Specify Initial Conditions (Model > Initial Conditions)

Initial conditions are generally associated with transient model runs. Their purpose is to provide a reasonable starting point for the solver. In a steady-state model (such as the model at hand), initial conditions can be used to "precondition" the solver to allow faster convergence. However, this "pre conditioning" is unnecessary for the current model. Therefore, the user should set the initial conditions to "None" using the following steps.

1. Select *Model* > *Initial Conditions* > *Settings*,
2. Select the "None" option,
3. Press *OK* to close the dialog.

d. Specify Boundary Conditions (Model > Boundaries)

Boundary conditions must be applied to region points. Once a boundary condition is applied to a boundary point the starting point is defined for that particular boundary condition. The boundary condition will then extend over subsequent line segments around the edge of the region in the direction in which the region shape was originally entered. Boundary conditions remain in effect around a shape until re-defined. The user cannot define two different boundary conditions over the same line segment.

More information on boundary conditions can be found in the *Menu System > Model Menu > Boundary Conditions > 2D Boundary Conditions* in your User's Manual.

Now that all of the regions, and surfaces have been successfully defined, the next step is to specify the boundary conditions on the region shapes. A number of heads will be defined on the Slope region segments, with the Zero Flux condition being applied to the remainder. The Reservoir will be set to have a head of 10.5m as a Surface 2 boundary condition. The steps for specifying the boundary conditions include:

1. Make sure your model is being viewed in 2D. To do this, click on the *2D* button found to the left side of the workspace,
2. Select "Slope" in the Region Selector,
3. Select "Surface 1" in the Surface Selector, found at the top of the workspace,
4. From the menu select *Model > Boundaries > Boundary Conditions*. The *boundary conditions* dialog will open and display the boundary conditions for Surface 1. These boundary conditions will extend from Surface 1 to Surface 2 over Layer 1. Select the *Boundary Conditions* tab. Select the first point (0,0) and assign a zero flux boundary condition by using the Boundary Condition drop-down,
5. Select the point (14,0) from the list,
6. From the Boundary Condition drop-down select a Head Expression boundary condition. This will cause the Constant/Expression box to be enabled,
7. In the Constant/Expression box enter a head of 7,
8. Select the point (14,27) from the list,
9. From the Boundary Condition drop-down select a Zero Flux boundary condition,
10. Press the *OK* button to close the dialog.

Slope Region Boundary Condition Summary

X	Y	Boundary Condition	Expression
0	0	Zero Flux	
14	0	Head Expression	7
24	0	Continue	
24	27	Continue	
14	27	Zero Flux	
0	27	Continue	

NOTE:
The Head Expression boundary condition for the point (14,0) becomes the boundary condition for the following line segments that have a Continue boundary condition until a new boundary condition is specified. In this case the line segments from (14,0) to (14,27) are all given a Head Expression boundary condition.

In order to set the Reservoir’s Surface 2 Boundary Condition to 10.5:

1. Select the “Reservoir” region in the Region Selector,
2. From the menu select *Model > Boundaries > Boundary Conditions,*
3. Select "Surface 2" from the surface drop-down,

4. Click the Surface Boundary Conditions tab,
5. Select the "Head Expression" boundary condition, and enter a value of 10.5 in the Constant/Expression box,
6. Press the *OK* button to close the dialog.

e. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material property for the material that will be used in the model. It will be defined for both the slope and reservoir regions.

1. Open the *Materials Manager* dialog by selecting *Model > Materials > Manager* from the menu,
2. Click the *New* button to create a material,
3. Enter "Tutorial Soil" into the Material Name field,
4. Set Data Type to Unsaturated,
5. Press *OK* to close the dialog. The *Material Properties* dialog will open automatically,

> **NOTE:**
> When a new material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned to it will display that material's fill color.

6. Under the *Volumetric Water Content* tab enter a Saturated VWC value of 0.4,
7. Move to the *Hydraulic Conductivity* tab,
8. Enter the ksat value of 2.512E-06 m/s,
9. Select "None" from the Unsaturated Hydraulic Conductivity option group,
10. Press *OK* on both dialogs to return to the workspace.

Each region will cut through all the layers in a model, creating a separate “block” on each layer. Each block can be assigned a material or be left as void. A void area is essentially air space. In this model all blocks will be assigned a material.

1. Select "Slope" in the Region Selector,
2. Select *Model > Materials > Material Layers* from the menu to open the *Material Layers* dialog,
3. Select "Tutorial Soil" from the drop-down for Layer 1,
4. Select the "Reservoir" region by using the right arrow button,
5. Select the "Tutorial Soil" material from the drop-down for Layer 1,
6. Close the dialog using the *OK* button.

f. Specify Model Output

Two levels of output may be specified: i) output (graphs, contour plots, fluxes, etc.) which are displayed during model solution, and ii) output which is written to a standard finite element file for viewing with ACUMESH software. Output is specified in the following two dialogs in the software:

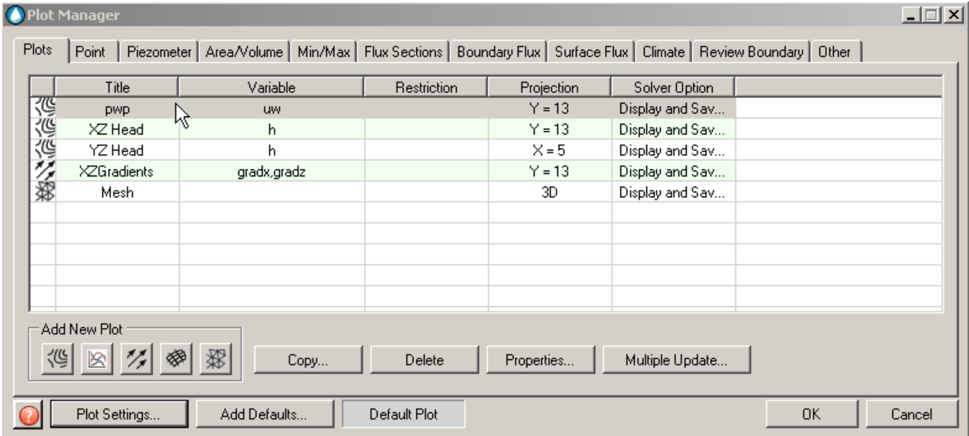
i) Plot Manager: Output displayed during model solution.
ii) Output Manager: Standard finite element files written out for visualization in ACUMESH or for inputting to other finite element packages.

PLOT MANAGER **(Model > Reporting > Plot Manager)**

Now that the model is defined, it is necessary to specify the reports which will be created by the solver such that the desired output from the model can be seen.

There are many plot types that can be specified to visualize the results of the model. A few will be generated for this tutorial example model including a plot of the pressure contours, head contours, solution mesh, and gradient vectors.

1. Open the *Plot Manager* dialog by selecting *Model > Reporting > Plot Manager* from the menu,



2. The toolbar at the bottom left corner of the dialog contains a button for each plot type. Click on the *Contour* button to begin adding the first contour plot. The *Plot Properties* dialog will open,
3. Enter the title pwp,
4. Select "uw" as the variable to plot from the drop-down,
5. Move to the Projection tab,
6. Select "Plane" as the Projection Option,
7. Select *Y* from the Coordinate Direction drop-down,
8. Enter 13 in the Coordinate field. This will generate a 2D slice at *Y* = 13m on which the pore water pressures will be plotted,
9. Click *OK* to close the *Plot Properties* dialog and add the plot to the list. The *Plot Manager* dialog can remain open,
10. Repeat steps 2 to 9 to create the plots as shown in the screen shot above. Note that the Vector and Mesh plots do not require the entry of a variable,
11. Click *OK* to close the *Plot Manager* and return to the workspace.

Features are used in this model to control mesh density for increased resolution at

selected slices.

To add the features:

1. Select the "Reservoir" region in the region selector,
2. Select *Draw > Model Geometry > Feature Polyline* from the menu,
3. With the mouse click on the point (0,11). (The SNAP and GRID options must be on in the workspace),
4. Click on the point (2,11),
5. Click on the point (2,16),
6. Double-click on the point (0,16) to finish the Feature.

g. Run Model (Solve > Analyze)

The next step is to analyze the model. Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model.

h. Visualize Results (Window > AcuMesh)

The visual results for the current model may be examined by selecting the *Window > ACUMESH* menu option.

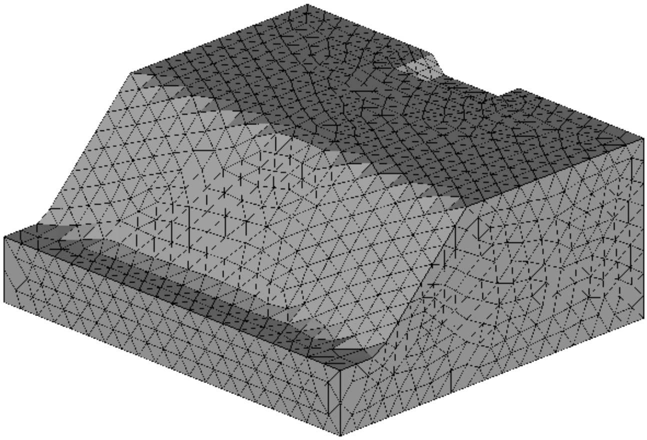
5.2 Results and Discussions

After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the FlexPDE solver. Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots. This section will give a brief analysis for each plot that was generated.

These reports are intended to provide the user with low-quality graphs which give a rough indication of the results. Creating professional-quality visualizations of the results can be accomplished with ACUMESH software, the use of which is described in the following section.

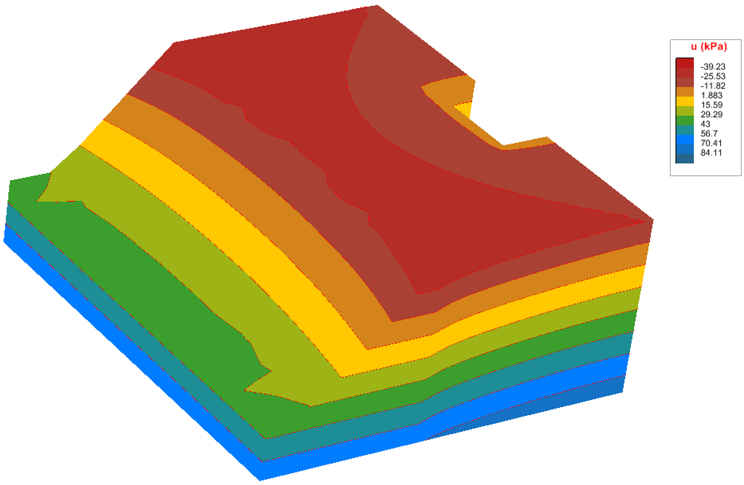
- **Solution Mesh**

The Mesh plot displays the finite-element mesh generated by the solver. The mesh is automatically refined in critical areas. Right-click on the plot and select "Rotate" to enable the rotate window.



- **Pressure Contours**

The most important contour in the above plot is the one that corresponds to zero pressure. This contour represents the phreatic surface. All material that lies below this line is saturated and all material that lies above this line is considered to be unsaturated. The plot indicates a gradual decrease in water pressure from the left to the right in the plot.



- **Head Contours**

As expected, the head is 10.5m at the left of the plot where this condition was specified as a boundary condition. The head decreases to 7m on the left edge of the plot where the boundary condition was set to 7m.

- **Flow Vectors**

Flow Vectors show both the direction and the magnitude of the flow at specific points in the model. Vectors illustrate that flow is from left to right in this view.

Once the model has been analyzed with the FlexPDE solver, it can be visualized using the ACUMESH software. ACUMESH will read a .DAT file, which can be defined in the Output Manager as described in previous sections of this tutorial. Once the .DAT file has been created the model can be visualized by clicking on the AcuMesh icon on the process toolbar.

The uncolored mesh should now be displayed in ACUMESH. In order to display contours of pressure head the user must:

1. Select *Plot* > *Contours* from the menu,
2. Under the Regions tab, choose "Show All",
3. Select 'uw' as the Variable Name, and
4. Press *OK* to close the dialog.

The mesh can be turned off for certain regions through the following process:

1. Select *Plot* > *Mesh* from the menu,
2. Select "Slope" in the list and uncheck the checkbox, and
3. Press *OK* to close the dialog.

Flow vectors can be displayed through the following process:

1. Select *Plot* > *Vectors* from the menu,
2. Select "Resevoir" in the list and ensure the checkbox is checked, and
3. Press *OK* to close the dialog.

The current visualization can be exported on a standardized format through the following steps:

1. Select *File* > *Export As* from the menu,
2. Select a file name, and
3. Specify a file type.

5.3 Model Data

Surface Data for Tutorial – Reservoir3D:

Surface 2

X	Y	Z
0	0	11
0	10	11
0	11	10
0	16	10
0	17	11
0	27	11
2	0	11
2	10	11
2	11	10
2	16	10
2	17	11
2	27	11

3	0	11
3	10	11
3	11	11
3	16	11
3	17	11
3	27	11
14	0	11
14	10	11
14	11	11
14	16	11
14	17	11
14	27	11
21	0	4
21	10	4
21	11	4
21	16	4
21	17	4
21	27	4
24	0	4
24	10	4
24	11	4
24	16	4
24	17	4
24	27	4

Region shape data for Tutorial – Reservoir3D:

Slope Region			Reservoir Region	
X	**Y**		**X**	**Y**
0	0		0	10
14	0		3	10
24	0		3	17
24	27		0	17
14	27			
0	27			

6 3D Planar Geometry

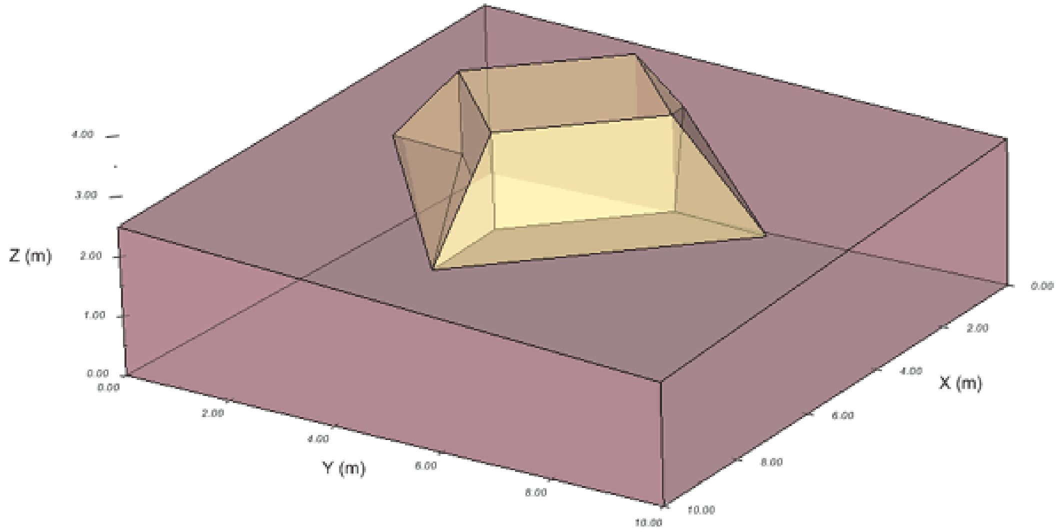
This example demonstrates the use of the new planar geometry feature in SVOFFICE in the creation of a tailings pile. Planar geometry allows the user to specify surface data using angular measurements instead of converting them to tables of grid points. This can greatly simplify model creation.

In this tutorial, a non-level ground plane will be created, and a simple truncated pyramid will be added to model the tailings pile above it. A built-in feature will be used to calculate the intersection between the two objects.

Project: MineTailings
Model: PlanarIntersection
Minimum authorization required: STUDENT

Model Description and Geometry

A simple 10m by 10m area is created. A non-level plane is added to model the ground surface. A pyramid-shaped tailings pile is added, with sidewalls at 45 degree angles to a flat ground surface. The intersection between the pile and the sloping ground surface is then determined by the software. The top of the tailings pile is set at 4m.



6.1 Model Setup

The following steps will be required to set up the model:

a. Create model
b. Enter geometry
c. Specify initial conditions
d. Specify boundary conditions
e. Apply material properties
f. Specify model output

g. Run model
h. Visualize results

a. Create Model

The first step in defining a model is to select or create a project for the new model. For this example, we will assume a project called "UserTutorial".

To add a new project, follow these steps:

1. Access the *SVOFFICE Manager* dialog. Skip to the next section if the project "UserTutorial" already exists,
2. Click *New* in the upper right of the Projects section,
3. The *Create New Project* dialog is opened along with a prompt asking for a new Project Name,
4. Type "UserTutorial" as the new Project Name and press *OK,*

The *Project Properties* dialog is where information specific to each project is stored. This will include the Project Name, Project Folder, and Project Notes information.

> **NOTE:**
> The Project Name is the only required information needed to define a project. The Project Name must be unique; the rest of the fields are optional.

5. Fill out the dialog with the desired information,
6. Click *OK* to exit this dialog and return to the *SVOFFICE Manager*. The new project will appear in the Projects list.

To add a model to the UserTutorial project follow these steps:

1. Select the "UserTutorial" project from the Project Name list,
2. Click *New* in the upper right of the Models section,
3. The *Create New Model* dialog is opened along with a prompt asking for a new Model Name,
4. Enter PlanarIntersection in the Model Name box,
5. Select the following:

Application:	SVFLUX	
System:	3D	
Type:	Steady-State	
Units:	Metric	
Time Units:	Day (d)	
World Coordinate System:		
xmin = 0	ymin = 0	zmin = 0
xmax = 10	ymax = 10	zmax = 4

6. Click the *OK* button to save the model and close the *New Model* dialog,
7. The new model will automatically be opened in the workspace.

The workspace grid spacing aids in defining region shapes. The default spacing is fine for this model, simply close the dialog by pressing *OK*.

b. Enter Geometry (Model > Geometry)

A region in SVFLUX is the basic building block for a model. A region represents both a physical portion of material being modeled and a visualization area in the SVFLUX CAD workspace. A region will have a set of geometric shapes that define its material boundaries. Also, other modeling objects including features, flux sections, water tables, text, and line art can be defined on any given region.

This model will be divided into six regions, which are named Extents, Top, North, South, East and West. To add the necessary regions follow these steps:

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions* from the menu,
2. Change the first region name from R1 to Extents. To do this, highlight the name and type the new text,
3. Press the *New* button to add a second region,
4. Change the name of the second region to Top,
5. Repeat steps 3 and 4 to add regions for North, East, South, and West,
6. Click *OK* to close the dialog.

The shapes that define each region will now be created. Note that when drawing geometry shapes, the region that is current in the Region Selector is the region to which the geometry will be added. The Region Selector is at the top of the workspace. Refer to the 3D Planar Geometry Model Data section for the geometry points for each region.

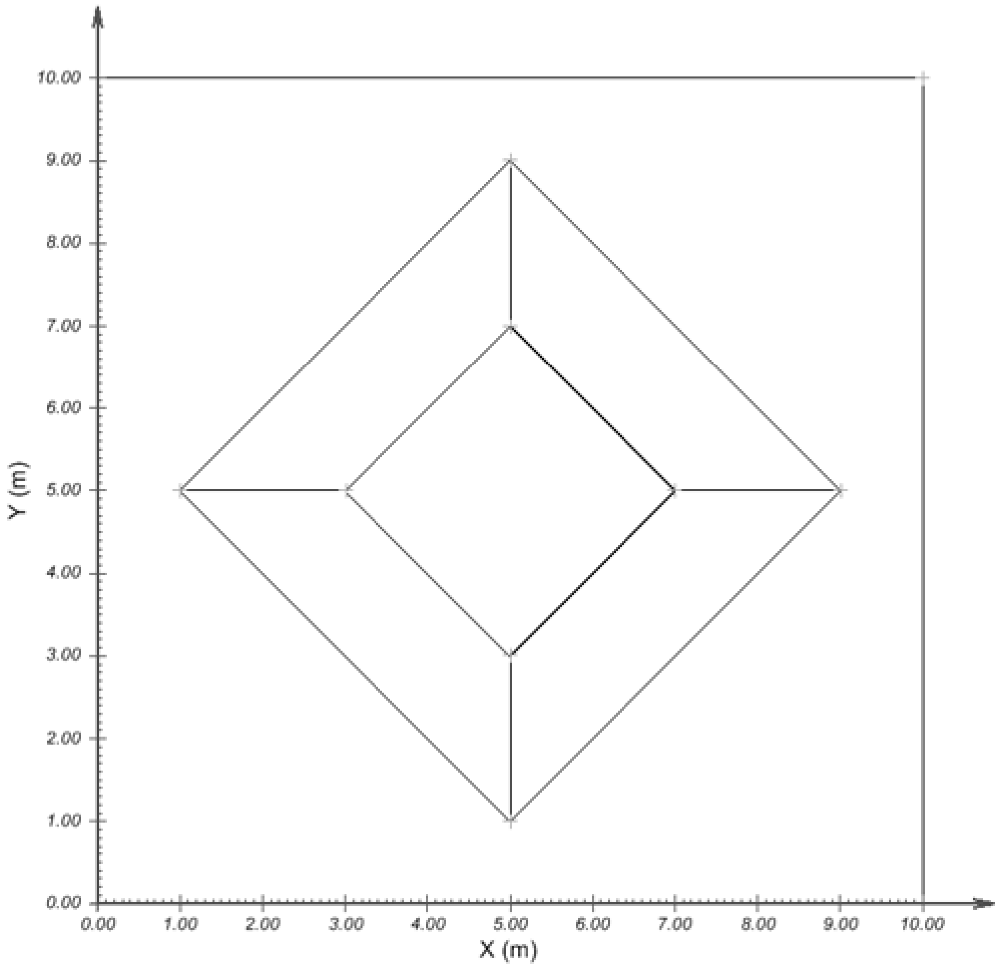
- **Define the Extents region**

1. Ensure that the SNAP and GRID options in the status bar are both On. These options ensure that your data points are properly aligned,
2. Select "Extents" in the Region Selector, found in the toolbar at the top of the workspace,
3. Select *Draw > Model Geometry > Region Polygon* from the menu,
4. Move the cursor near (0,0) in the drawing space. You can view the coordinates of the mouse's current position in the status bar just below the drawing space,
5. Click once with the left mouse button near (0,0) to create the first region point,
6. Now move the cursor near (0,10) and left click on the point. A line is now drawn from (0,0) to (0,10),
7. Repeat step 6 for the following coordinates: (10,10), (10,0),
8. For the final point, (10,0), double-click on the point to finish the shape. SVFLUX completes the shape automatically by drawing a line from (10,0) to (0,0).

NOTE:
If a mistake was made entering the coordinate points for a shape, edit the shape using the *Region Properties* dialog (menu item *Model > Geometry > Region Properties*) or double-click on the shape just drawn.

- **Define the inner regions (Top, North, East, South and West)**

9. Select the region "Top" in the Region Selector,
10. Select *Draw > Model Geometry > Region Polygon* from the menu,
11. Using the "Region: Top" table in the Model Data section below, move the cursor near each point in the order given, and click once to add that point. After adding the final point, double-click on that point to finish the shape.
12. Repeat the previous three steps for the North, East, South and West regions. Each region has a "Region: <name>" table in the Model Data section.
13. Your geometry should now look like the following:



Every 3D model has two surfaces by default. Our model has three surfaces, so we must add one additional surface before defining the third one.

- **Define Surface 1**

The bottom surface is a constant elevation, and is the simplest to define.

1. Select "Surface 1" in the Surface Selector found at the top of the workspace,
2. Select *Model > Geometry > Surface Properties* from the menu to open the *Surface Properties* dialog,
3. For the Surface Definition Option, select Constant from the drop-down,
4. Click on the Constant tab,
5. Enter an Elevation of 0,
6. Click *OK* to close the dialog,

- **Define Surface 2**

The ground surface is defined as a single non-level plane. We will enter this plane once and apply it to every region in the model.

7. Select "Surface 2" in the Surface Selector found at the top of the workspace,
8. Select *Model > Geometry > Surface Properties* from the menu to open the *Surface Properties* dialog,
9. For the Surface Definition Option, select Plane from the drop-down,
10. Select "Extents" from the Region list,
11. Using the "Surface 2: All Regions" table from the Model Data section, enter the Z values in the grid on the right. The X and Y values should already be present, but they can be edited if necessary,
12. Highlight the nine values by clicking on the top-left value, and then clicking on the bottom-right value while holding down the Shift key,
13. Press Ctrl-C to copy the values,
14. Select "Top" from the Region list,
15. Press the *Paste* button to retrieve the same values entered above,
16. Repeat the previous two steps for every region in the list,
17. Click *OK* to close the dialog,

- **Define Surface 3**

The upper surface is defined as a series of interconnected planes. Each region will have a unique plane. The planes are entered using simple geometry, and then a built-in calculator will determine the correct intersection points for the geometry.

IMPORTANT NOTE:
It can be complicated to calculate intersection points for 3D shapes. It is usually best to plan ahead and simplify your geometry as much as possible to ensure ease of entry. The built-in calculator in this example removes the need to calculate the exact intersection points between the truncated pyramid shape and the non-level ground plane. All that is required is that the planar geometry entered by the user is accurate before attempting to use this feature.

More complex intersections are also possible, and can be calculated by using the

general-purpose planar intersection calculator found in the *Model > Geometry > Tools > Plane Intersection* menu item.

18. Go to *Model > Geometry > Surfaces* in the menu to open the *Surfaces* dialog,
19. Press the "New..." button to add a new surface. Accept the defaults by pressing OK on the *Insert Surfaces* dialog,
20. Press *OK* to close the *Surfaces* dialog,
21. Select "Surface 3" in the Surface Selector found at the top of the workspace,
22. Select *Model > Geometry > Surface Properties* from the menu to open the *Surface Properties* dialog,
23. For the Surface Definition Option, select Plane from the drop-down,
24. Select "Extents" from the Region list,
25. Ensure all Z values for this region are set to 0, as this region has no surface data on Surface 3,
26. Select "Top" from the Region list,
27. Using the "Surface 3: Region Top" table from the Model Data section, enter the Z values in the grid on the right. The X and Y values should already be present, but they can be edited if necessary,
28. Select "North" from the Region list,
29. Using the "Surface 3: Region North" table from the Model Data section, enter the Z values in the grid on the right. The X and Y values should already be present, but they can be edited if necessary,
30. Press the *Calculate Plane-Plane Intersection* button to calculate the correct model intersection points,
31. Repeat the previous three steps for the East, South, and West regions,
32. Press *OK* to close the dialog,

The "Extents" region must be marked as Limited as it has no surface data above the ground surface (Surface 2) to complete the geometry. In addition, all other regions must be marked as Limited below the ground surface.

22. Select "Extents" in the Region Selector found at the top of the workspace,
23. Select *Model > Geometry > Region Polygon* from the menu,
24. Press the *Limited Region...* button,
25. Select "Layer 2" from the list and press the *Exclude* button,
26. Press *OK* to close the *Limited Region* dialog,
27. Press the right arrow button in the top right of the *Region Properties* dialog to advance to the "Top" region,
28. Press the *Limited Region...* button,
29. Select "Layer 1" from the list and press the *Exclude* button,
30. Press *OK* to close the *Limited Region* dialog,
31. Repeat the previous four steps for the North, East, South and West regions,

32. Press *OK* to close the dialog.

The model should now appear similar to the image displayed in the 3D Planar Geometry section above. Use the *3D* button on the left side to open the 3D view.

c. Specify Initial Conditions (Model > Initial Conditions)

Initial conditions are generally associated with transient model runs. Their purpose is to provide a reasonable starting point for the solver. In a steady-state model (such as the model at hand), initial conditions can be used to "precondition" the solver to allow faster convergence.

1. Select *Model > Initial Conditions > Settings*,
2. Select the "Head Constant/Expression" option,
3. Enter a Head of 1,
4. Press *OK* to close the dialog.

d. Specify Boundary Conditions (Model > Boundaries)

Boundary conditions must be applied to region points. Once a boundary condition is applied to a boundary point, the starting point is defined for that particular boundary condition. The boundary condition will then extend over subsequent line segments around the edge of the region in the direction in which the region shape was originally entered. Boundary conditions remain in effect around a shape until re-defined. The user cannot define two different boundary conditions over the same line segment.

More information on boundary conditions can be found in the *Menu System > Model Menu > Boundary Conditions > 2D Boundary Conditions* in your User's Manual.

Now that all of the regions, and surfaces have been successfully defined, the next step is to specify the boundary conditions on the region shapes. A head will be applied to some of the Extents region, along with an area of zero flux to force the flow in a specific direction. A surface boundary condition will be applied to the top surface of the model to simulate a continuous, constant light rainfall.

1. Select "Extents" in the Region Selector,
2. Select "Surface 1" in the Surface Selector, found at the top of the workspace,
3. From the menu select *Model > Boundaries > Boundary Conditions*. The *Boundary Conditions* dialog will open and display the boundary conditions for Surface 1,
4. Select the point (0,0) from the list,
5. From the Boundary Condition drop-down select a Zero Flux boundary condition,
6. Select the point (0,10) from the list,
7. From the Boundary Condition drop-down select a Head Expression boundary condition. This will cause the Constant/Expression box to be enabled,
8. In the Constant/Expression box enter a head of 1,
9. Select the point (10,10) from the list,
10. From the Boundary Condition drop-down select a Zero Flux boundary condition,
11. Select the point (10,0) from the list,
12. From the Boundary Condition drop-down select a Head Expression boundary

condition. This will cause the Constant/Expression box to be enabled,

13. In the Constant/Expression box enter a head of 1,
14. Click on the *Surface Boundary Conditions* tab,
15. Ensure that the Boundary Condition is set to None,
16. Select "Surface 2" from the Surface list,
17. From the Boundary Condition drop-down select a Z-Flux Expression boundary condition,
18. In the Constant/Expression box enter a value of 0.0003,
19. Press the *OK* button to close the dialog,

Each of the five top surfaces now requires the same z-flux expression listed above.

20. Select "Top" in the Region Selector,
21. Select "Surface 3" in the Surface Selector, found at the top of the workspace,
22. From the menu select *Model > Boundaries > Boundary Conditions*. The *Boundary Conditions* dialog will open and display the boundary conditions for Surface 3,
23. Click on the *Surface Boundary Conditions* tab,
24. From the Boundary Condition drop-down select a Z-Flux Expression boundary condition,
25. In the Constant/Expression box enter a value of 0.0003,
26. Press the *OK* button to close the dialog,
27. Repeat the previous seven steps for the North, East, South, and West regions.

The Boundary Conditions display on the screen should now look similar to the following:

Boundary Conditions

- Extents, Surface 1 (0, 10), Head Expression (1 m), BN7
- Extents, Surface 1 (10, 0), Head Expression (1 m), BN9
- Extents, Surface 2, Z-Flux Expression (0.0003 m^3/day/m^2)
- Top, Surface 3, Z-Flux Expression (0.0003 m^3/day/m^2)
- North, Surface 3, Z-Flux Expression (0.0003 m^3/day/m^2)
- East, Surface 3, Z-Flux Expression (0.0003 m^3/day/m^2)
- South, Surface 3, Z-Flux Expression (0.0003 m^3/day/m^2)
- West, Surface 3, Z-Flux Expression (0.0003 m^3/day/m^2)

If there are additional boundary conditions defined, they will need to be removed. Use the menu item *Model > Boundaries > Boundary Conditions* menu to select the *Boundary Conditions* dialog and correct the problem.

e. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the materials that will be used in the model.

1. Open the *Materials Manager* dialog by selecting *Model > Materials > Manager* from the menu,
2. Click the *New* button to create a material,

3. Enter "Ground" into the Material Name field,
4. Set Data Type to Unsaturated,
5. Press *OK* to close the dialog. The *Material Properties* dialog will open automatically,

NOTE:
When a new material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned to it will display that material's fill color.

6. Under the *SWCC* tab enter a Saturated VWC value of 0.5,
7. Move to the *Hydraulic Conductivity* tab,
8. Enter the ksat value of 8.32E-04 m/day,
9. Select "None" from the Unsaturated Hydraulic Conductivity option group,
10. Return to the *SWCC* tab,
11. Select "Fredlund and Xing Fit" from the SWCC option group,
12. Press *Properties...* from the SWCC option group to access the *Fredlund and Xing Fit* dialog,
13. Enter the following values in their respective fields: af=10, nf=1, mf=1.4, hr=3000,
14. Press *OK* twice to return to the *Materials Manager*,
15. Click the *New* button to create a second material,
16. Enter "Pile" into the Material Name field,
17. Set Data Type to Unsaturated,
18. Press *OK* to close the dialog. The *Material Properties* dialog will open automatically,
19. Under the *SWCC* tab enter a Saturated VWC value of 0.5,
20. Move to the *Hydraulic Conductivity* tab,
21. Enter the ksat value of 4.32E-04 m/day,
22. Select "None" from the Unsaturated Hydraulic Conductivity option group,
23. Return to the *SWCC* tab,
24. Select "Fredlund and Xing Fit" from the SWCC option group,
25. Press *Properties...* from the SWCC option group to access the *Fredlund and Xing Fit* dialog,
26. Enter the following values in their respective fields: af=30, nf=0.8, mf=1.1, hr=3000,
27. Press *OK* twice to return to the *Materials Manager*,
28. Press *OK* one more time to close the *Materials Manager* and return to the workspace.

Now that the materials are defined, they can be assigned to the region layers. Region layers

that are set as Limited should be left as void (which is the default). A void area is essentially air space.

1. Select "Extents" in the Region Selector,
2. Select *Model > Materials > Material Layers* from the menu to open the *Material Layers* dialog,
3. Select "Ground" from the drop-down for Layer 1 (leave Layer 2 as void),
4. Press the right arrow at the top right of the dialog to select the "Top" region,
5. Select "Pile" from the drop-down for Layer 2 (leave Layer 1 as void),
6. Repeat the previous two steps for the North, East, South and West regions,
7. Press *OK* to close the dialog.

f. Specify Model Output

Two levels of output may be specified for a model: i) output (graphs, contour plots, fluxes, etc.) which are displayed during model solution, and ii) output which is written to a standard finite element file for viewing with ACUMESH software. Output is specified in the following two dialogs in the software:

i) Plot Manager:	Output displayed during model solution.
ii) Output Manager:	Standard finite element files written out for visualization in ACUMESH or for inputting to other finite element packages.

SVFLUX uses a "smart" output system that pre-defines outputs for you automatically when the model is analyzed. In this tutorial, no special plots are required so no extra steps are required here.

g. Run Model (Solve > Analyze)

The next step is to analyze the model. Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model.

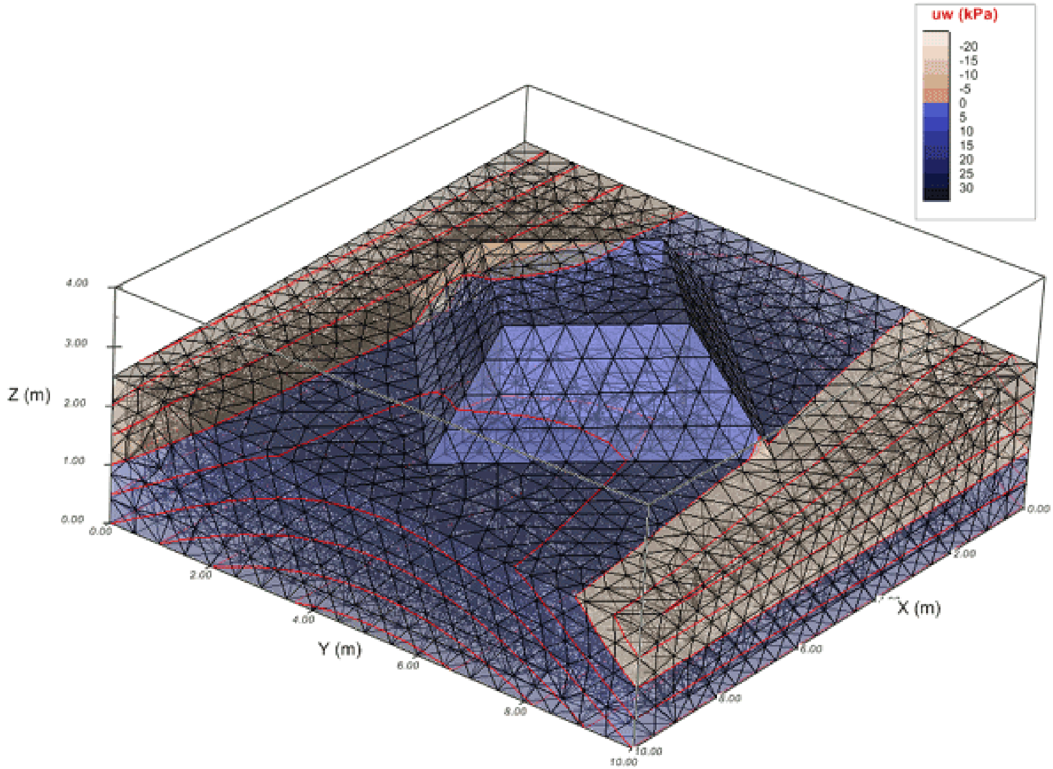
h. Visualize Results (Window > AcuMesh)

The visual results for the current model may be examined by selecting the *Window > AcuMesh* menu option.

6.2 Results and Discussions

As with the other tutorials, preliminary examination of the results can be done by examining the plots produced in FlexPDE. A variety of default plots is available.

Planar models generally run very quickly. The final output can be viewed in AcuMesh by pressing the AcuMesh icon or from the Window > AcuMesh menu item. It should appear similar to the following:



As with the other tutorials, many options are available to visualize this model. A detailed list is not included here, as the purpose of this model is to demonstrate planar geometry. Please refer to our other tutorials for more information about AcuMesh.

6.3 Model Data

Region shape data for Tutorial – PlanarIntersection:

Region: Extents			Region: Top	
X	Y		X	Y
0	0		3	5
0	10		5	7
10	10		7	5
10	0		5	3

Region: North			Region: East			Region: South			Region: West	
X	Y		X	Y		X	Y		X	Y
7	5		5	3		5	3		1	5
5	7		7	5		3	5		3	5
5	9		9	5		1	5		5	7
9	5		5	1		5	1		5	9

Surface Data for Tutorial – PlanarIntersection:

Surface 2: All Regions

X	Y	Z
0	0	3
0	10	2.5
10	10	2

Surface 3, Region: Extents **Surface 3, Region: Top**

X	Y	Z		X	Y	Z
0	0	0		3	5	4
0	10	0		5	7	4
10	10	0		7	5	4

Surface 3, Region: North **Surface 3, Region: East** **Surface 3, Region: South** **Surface 3, Region: West**

X	Y	Z		X	Y	Z		X	Y	Z		X	Y	Z
7	5	4		5	3	4		5	3	4		1	5	2
5	7	4		7	5	4		3	5	4		3	5	4
5	9	2		9	5	2		1	5	2		5	7	4

7 References

FlexPDE 6.x Reference Manual, 2007. PDE Solutions Inc. Spokane Valley, WA 99206.

Fredlund, D. G., and Xing, A., 1994, Equations for the soil-water characteristic curve, Canadian Geotechnical Journal, Vol. 31, No. 3, pp. 521-532.

This page is left blank intentionally.